

University Turbine Systems Research (UTSR) Industrial Fellowship Program

Next Generation Gas Turbine (NGGT) Aero Rig

Prepared By:

Juan Hincapie
Department of Mechanical Engineering,
University of Louisiana at Lafayette
Lafayette, LA 70503

Prepared For:



South Carolina Institute for Energy Studies (SCIES)
400 Klugh Ave.
Clemson University
Clemson, SC 29634-5712



General Electric Company (GE Energy)
Gas Turbine Aero
Greenville, SC 29615-5102

Abstract

General Electric – Energy (GE Energy) is in the process of building a state-of-the-art industrial gas turbine. In order to study technologies that can increase overall turbine efficiency and validate the computational fluid dynamics (CFD) code used to design gas turbines, a 1:5 scale turbine aero rig of an actual power generation turbine is designed, built, and tested. This paper presents and discusses the data reduction, data interpretation, and CFD model analysis of the 3-stage power generation aero turbine rig that was run for a wide range of operating conditions. The rig is equipped with multiple sensors to measure pressure (total and static), temperature (total and static), mass flow, torque, and speed. A data reduction tool is then used to determine mixed-out traverse profiles and to generate contour plots of pressure, temperature, velocity, and swirl at different locations of the flowpath. The interpretation of such results serves to validate the CFD models used in the current design practice and to determine how new technologies impact the engine's efficiency, which determines the operating cost of the engine.

1. Introduction

In industrial gas turbine applications, CFD probably plays a greater part in the aerodynamic design of turbomachinery than in any other engineering applications. In general, the design process is performed using CFD and other 1D and 2D computer-aided tools. The actual performance of the engine is not known until the first full-scale prototype is built since the design of scaled prototypes is too expensive and time consuming. The design of turbines and compressors would be unthinkable without the help of CFD, which allows for shorter design cycles to better performance and reduced costs. GE Energy, with support for the Department of Energy (DOE), has designed and built the Next Generation Gas Turbine (NGGT) aero rig, which is a 1:5 scale of an actual power generation turbine for which field performance data is available over a wide range of operating conditions and is used as a benchmark in the company's design process.

Due to the proprietary nature of the technologies and the instrumentation used in the test of the NGGT aero rig, the different configurations and operating conditions of the test can't be shown or mentioned. The goal of the test is to: measure the performance of the turbine over a wide range of operating points; obtain detailed inter-stage data; and measure the performance of advanced technologies operating in a multi-stage turbine environment. The data collected from the test will be used to validate and verify advanced 3D CFD design tools and methodologies. In addition, the test will provide a solid baseline for future potential tests to develop technology such as advanced flow path geometries, 3D aero technologies, and advanced last stage bucket and diffuser systems.

In essence, the NGGT rig provides the validation data necessary to benchmark developed 3D turbine aerodynamic technologies enabling improvement in gas turbine efficiency and lower \$/kw operating costs.



2. Experimental System

The NGGT rig design consists of a fully cooled 1:5 scale of the turbine from an actual power generation turbine, as shown in Fig. 1. It is critical to understand and account for any hardware deviations from design intent. Any non-conformance in flowpath hardware can alter losses and therefore affect performance of the turbine. For all the airfoil rows, trailing edge thicknesses, throats, surface roughness, and surface profiles were measured and validated. It is important to note that the rig contains only the turbine section, omitting the compressor and combustor. The inlet flow is generated by a compressor situated at the testing facility and is heated in order to meet the inlet temperature requirement.

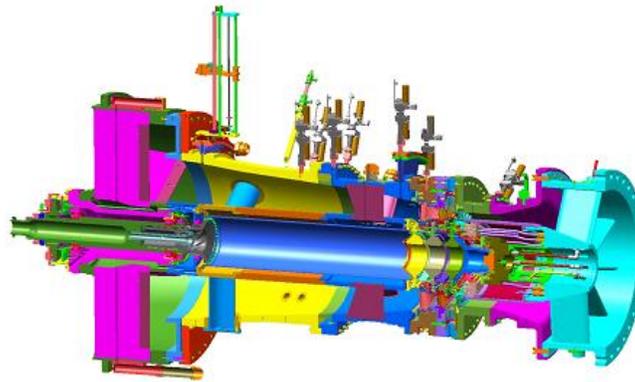


Figure 1. Model of the NGGT aero rig

The rig is heavily cooled, consisting of 16 internal secondary flow circuits that are independent, thereby allowing the distribution of cooling flows to airfoils, shrouds, and the wheelspace. Fig. 2 shows the assembled rig in the test cell. The aero similarity was maintained through the scaling to best match leakages and cooling mixing losses.

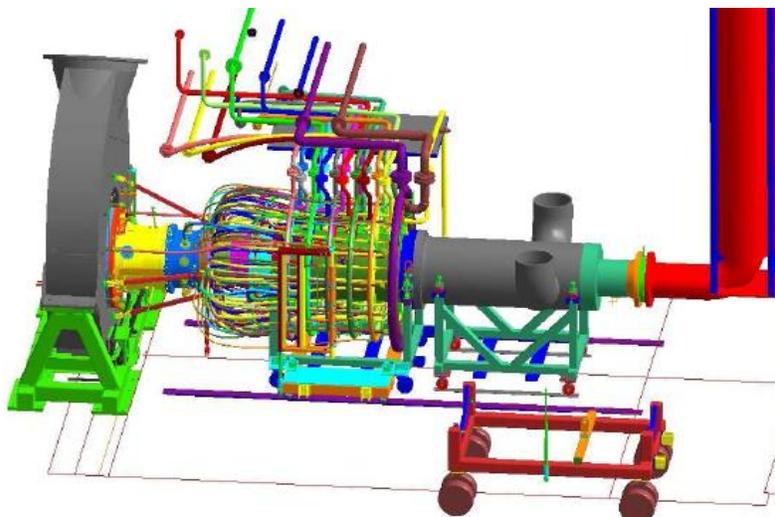


Figure 2. Assembly of NGGT rig in test cell



3. Experimental Data Reduction and Analysis

The traverse data reduction process is performed using a Matlab code. The code reads the data files generated by the data acquisition system and performs a constant area 2D mixing calculation to determine mixed-out profiles of pressure, temperature, swirl, and velocities at each of the sensor's locations, which are placed in key flowpath positions to determine engine performance. The 2D mixing plane approach calculates the mixing of the wake from the trailing edge to a point far downstream where uniform flow conditions exist.

The continuity equation relates the mass flux at station ∞ to that at station 2:

$$b\rho_{\infty}C_{2\infty} = \int_0^b \rho_2 C_{z2} du . \quad (1)$$

The tangential component of the momentum flux equation is:

$$b\rho_{\infty}C_{2\infty}C_{u\infty} = \int_0^b \rho_2 C_{z2} C_{u2} du . \quad (2)$$

The axial component of the momentum equation is:

$$b(p_{\infty} + \rho_{\infty}C_{z\infty}^2) = b p_2 + \int_0^b \rho_2 C_{z2} C_{z2}^2 du . \quad (3)$$

The energy equation can be written as:

$$C_p T_{\infty} + \frac{1}{2}(C_{u\infty}^2 + C_{z\infty}^2) = C_p \bar{T} T_2 . \quad (4)$$

The equation of state is:

$$p = \rho RT . \quad (5)$$

Equations (1) and (2) can be combined to give:

$$C_{u\infty} = \frac{\int_0^b \rho_2 C_{z2} C_{u2} du}{\int_0^b \rho_2 C_{z2} du} . \quad (6)$$



Now, take equation (4), eliminate T_∞ from it with equation (5), and eliminate $C_{z\infty}$ from it with equation (1). In addition, substitute equation (1) into equation (3) to eliminate $C_{z\infty}$. The resulting equation is used to eliminate p_∞ . Then, arrange algebraically to the form:

$$A\rho_\infty^2 - B\rho_\infty + C = 0 \quad (7)$$

where

$$A = C_p \bar{T} T_2 - \frac{1}{2} C_{u\infty}^2 \quad (8)$$

$$B = \frac{C_p}{R} \left(\bar{p}_2 + \frac{1}{b} \int_0^b \rho_2 C_{z2}^2 du \right) = \frac{\gamma}{\gamma-1} (\bar{p}_2 + I_{MZ}) \quad (9)$$

$$D = \left(\frac{C_p}{R} - \frac{1}{2} \right) \left(\frac{1}{b} \int_0^b \rho_2 C_{z2} du \right)^2 = \frac{\gamma+1}{2(\gamma-1)} (I_{CZ})^2. \quad (10)$$

The root equation (7) that applies for subsonic axial Mach number is:

$$\rho_\infty = \frac{1}{2A} (B + \sqrt{B^2 - 4AD}). \quad (11)$$

Using equation (11), the axial velocity can be found from equation (1):

$$C_{z\infty} = \frac{1}{\rho_\infty b} \int_0^b \rho_2 C_{z2} du = \frac{I_{CZ}}{\rho_\infty}. \quad (12)$$

The air angle is:

$$\beta_\infty = \tan^{-1} \left(\frac{C_{u\infty}}{C_{z\infty}} \right). \quad (13)$$

The static pressure is given by:

$$p_\infty = \bar{p}_2 + \frac{1}{b} \int_0^b \rho_2 C_{z2}^2 du - \frac{1}{\rho_\infty} \left(\frac{1}{b} \int_0^b \rho_2 C_{z2} du \right)^2 = \bar{p}_2 + I_{MZ} - \frac{1}{\rho_\infty} I_{CZ}^2. \quad (14)$$

The static temperature can be obtained from the equation of state:



$$T_{\infty} = \frac{P_{\infty}}{R\rho_{\infty}}. \quad (15)$$

The Mach number is:

$$M_{\infty} = \sqrt{\frac{C_{u\infty}^2 + C_{z\infty}^2}{\gamma RT_{\infty}}}. \quad (16)$$

Finally, the total pressure is obtained from the isentropic relationship:

$$PT_{\infty} = p_{\infty} \left(\frac{\bar{T}T_2}{T_{\infty}} \right)^{\frac{\gamma}{\gamma-1}}. \quad (17)$$

The Matlab code uses similar calculations as the ones previously shown and it also includes area and circumferential averaging techniques to determine the 2D mixing calculation. Fig. 3 depicts typical circumferentially averaged radial profiles generated by the Matlab code. Note that due to proprietary restrictions, the information shown is for illustrative purposes only.

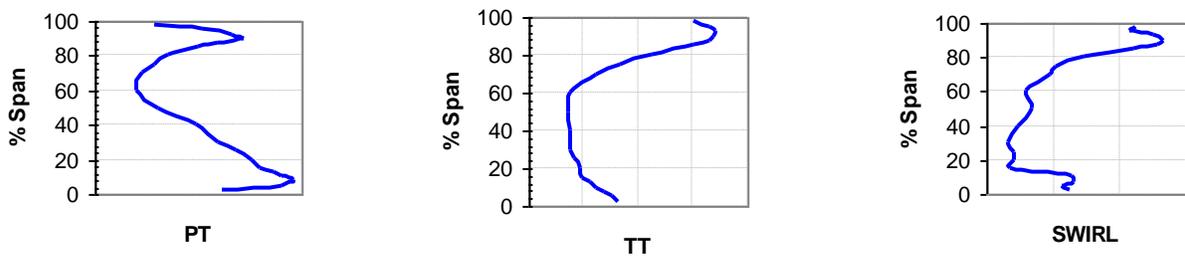


Figure 3. Mixed-out pressure, temperature, and swirl profiles generated by Matlab script

At GE Energy, the engineers in the Gas Turbine Aero Group use an in-house 3D CFD program called TACOMA to solve structured or unstructured grid non-linear and linear Euler/Navier-Stokes equations for turbomachinery blade rows. Using TACOMA, it is possible to evaluate mixed-out profiles of several parameters at any location of the flowpath. Comparing the results determined by TACOMA and those given by the test data will validate and verify the advanced 3D CFD design tool and methodologies. Fig. 4 illustrates the comparison for the results obtained in Fig. 3 to those given by a pre-test TACOMA model for the flowpath location.



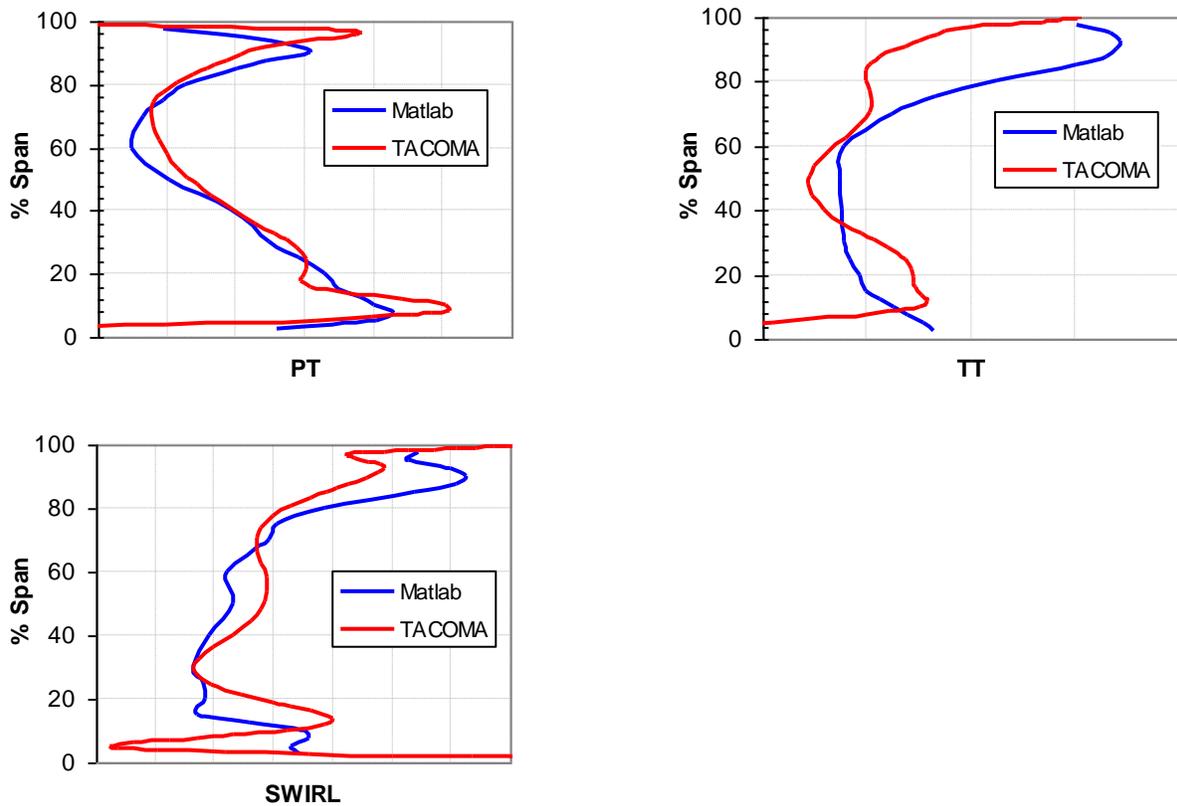


Figure 4. Comparison of mixed-out profiles – Matlab Vs. TACOMA

Fig. 4 depicts how the CFD program is modeling the flow physics in comparison to actual test data given by the NGGT rig. Since the TACOMA model ran was a pre-test prediction, many of the actual test conditions were not known at the time and influence the results. A close look at the pressure profile shows a similar relationship between both cases; however, there are some discrepancies at the hub and tip of the airfoil. Such differences might be caused by shock loss differences and secondary and/or cooling flows, which could be captured differently between the data and the CFD model. Additionally, pre-test assumptions, such as cooling, leakages, operating conditions, and actual geometries, made to run the TACOMA could be updated with the actual configuration that the rig was tested. Furthermore, the CFD program needs to be updated with correct flow levels and boundary conditions.

Examination of the temperature and swirl profiles in Fig. 4 portrays even higher discrepancies. The temperature profile is significantly different, which suggests that it's due to source term modeling. It is expected for discrepancies to significantly decrease as flow levels and cooling flow distributions are corrected. As for the swirl profile, changes in the pressure and temperature profiles will greatly affect the profile. Currently, the engineers are in the process of updating TACOMA with post-test conditions and redefining the source term modeling to match the secondary and cooling flows. Appendages, which more accurately model the wheelspaces, are also being added to account for different phenomenon that the CFD program can't accurately



model. The expectation is to match the CFD to the data and set a CFD model benchmark to be used for future gas turbine designs.

Another capability of the Matlab code, as well as TACOMA, is the ability to generate contour plots of pressure, temperature, velocity, and swirl at a specified location. Contour plots are a 2D representation of the flow parameter being evaluated at a specific location. The mixed-out profiles are created by averaging the contour plots; hence, the importance in being able to compare them graphically. Fig. 5 shows a graphical comparison of the pressure contour plot for the same conditions as Fig. 3 and 4.

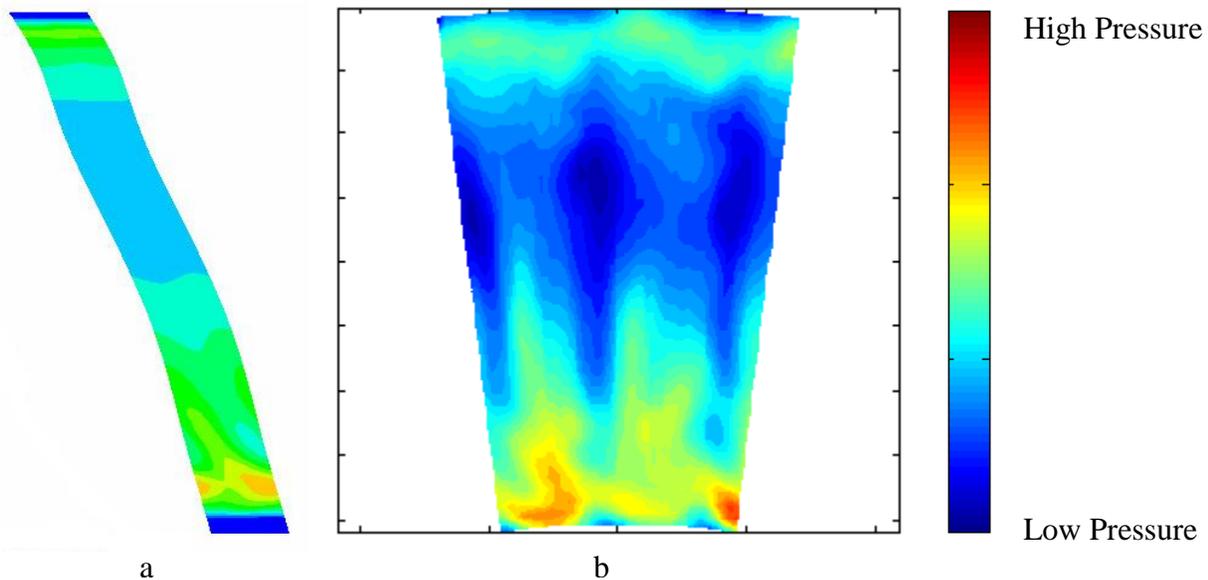


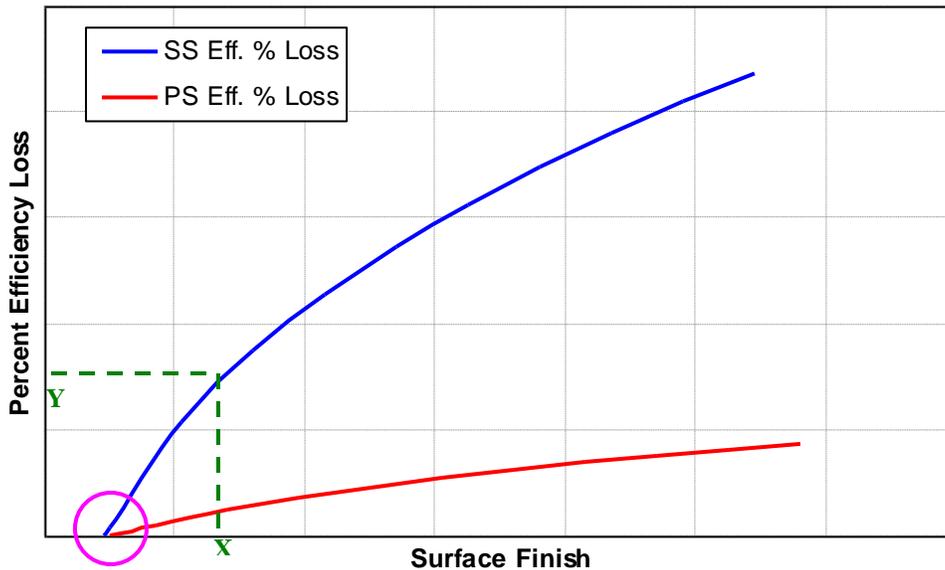
Figure 5. Comparison of PT contour – TACOMA (a) Vs. Matlab (b)

The contour plots are determined at different frames of reference; as a result, they are not portraying the same information. The Matlab contour plot is referenced from the nozzle and is an absolute plot while the TACOMA contour plot is referenced from the rotor and is a relative plot. Fig. 5 illustrates that the airfoil experiences an area of higher pressures at the hub, an area of lower pressures throughout the midspan, and an area of moderate pressure at the tip. Fig. 3 and Fig. 4 also portray such relationship.

One important parameter that is not modeled in the CFD model is the airfoil surface finish. The pre-test TACOMA assumes a hydraulically smooth roughness model. Airfoils are generally polished down to a specific surface roughness (RMS). Such roughness causes an increase in drag and generates efficiency losses due to surface friction. Using a computer-aided tool called SURFIN, it is possible to estimate a hydraulically smooth RMS, which represents the maximum roughness that causes no drag, much faster than incorporating a surface roughness model in TACOMA. This study is performed in both the pressure side and suction side of the airfoils. However, the suction side has a longer surface resulting in higher efficiency losses as well as higher Mach numbers.



Since it is difficult to scale RMS, a design target was set for the NGGT rig. The surface finish certainly affects losses but since the rig is a 1:5 scale and Reynolds numbers are lower, the effect will not be as drastic. Nevertheless, a study is conducted in SURFIN to determine how much loss in efficiency is incurred by manufacturing airfoils with a given RMS. Fig. 6 shows the result from the study for one of the airfoils of the rig.



Hydraulically Smooth Point

Figure 6. Estimated percent efficiency loss due to airfoil surface finish

The point at which the loss is zero set the hydraulically smooth RMS that is desired for the airfoil design. If the manufacturing process can't produce such RMS, then it is necessary to know how much efficiency is lost to adjust the CFD performance estimate. From Fig. 6, given a measured RMS of x , the loss in efficiency corresponds to y points in the evaluated bladerow efficiency. Typically, a small decrease in the overall turbine efficiency can increase the operating cost of an engine by thousands of dollars per hour. Therefore, monitoring the change in efficiency during the design of the engine using CFD and other computer-aided tools is of vital importance.

For the analysis of the rig performance, other technologies were also studied but can't be mentioned due to proprietary restrictions. In addition, during the duration of the UTSR fellowship, other assignments were performed in projects that are even more sensitive and thus are not discussed in this report.

4. Conclusions

The data reduction, data interpretation, and CFD model analysis of the NGGT 3-stage power generation aero turbine rig test conducted over the 11 week UTSR Fellowship at GE Energy concluded that:



1. The developed 1:5 scaled prototype of an actual power generation turbine and its instrumentation demonstrated good response to the testing operating conditions and were able to generate the data necessary to conduct research and validate and verify the advanced 3D CFD design tools and methodologies used to design gas turbines.
2. The analyzed data and CFD solution depict discrepancies that can be attributed to the modeling of source terms, boundary conditions, shock losses, leaks, and other phenomena. Hence the need to update the CFD model to validate it and provide a 3D CFD model benchmark.
3. Due to proprietary limitations, not all the work that was performed on the NGGT aero rig or other projects could be discussed in this paper.

5. Acronyms

CFD	computational fluid dynamics
DOE	department of energy
NGGT	next generation gas turbine
PS	pressure side
SS	suction side

6. Nomenclature

b	tangential blade spacing
C	flow velocity
C_p	specific heat capacity
M	mach number
p	static pressure
PT	total pressure
SWIRL	exit axial flow angle
T	static temperature
TT	total temperature
R	gas constant
RMS	surface roughness
B	air angle
γ	specific heat ratio
ρ	fluid density

Subscripts and superscripts

1	upstream
2	blade trailing edge
u	tangential direction
z	axial direction
∞	far downstream



- area average

7. Comments on UTSR Industrial Fellowship

The UTSR Fellowship gave me the opportunity to work with GE Energy, one of the world's leaders in the design of gas turbines. My main responsibility while at GE Energy was to work with the Gas Turbine Aero Group performing data reduction, data interpretation, and CFD model analysis of a 3-stage power generation turbine rig. I was also involved in other projects including modeling source terms and performing a bow and lean study on airfoils to see the effects on performance. All of the knowledge that I have learned throughout the 11-week fellowship has really focused my career objective. Once I am finished with my graduate studies, I will definitively seek employment or post-graduate studies in the gas turbine field.

I would like to thank SCIES and Gunnar Siden for providing me with the opportunity to be a part of the UTSR Industrial Fellowship Program at GE Energy and the whole Gas Turbine Aero Group for taking me in and providing me with their guidance. Special thanks to Neil Ristau for his assistance as a mentor and Ross Gustafson for taking me into his home and showing me a good time in Greenville.

